Continue

Robot structural analysis training pdf

Robot structural analysis price. Robot structural analysis 2022. Robot structural analysis tutorials. Autodesk robot structural analysis training manual pdf. Robot structural analysis forum. Robot structural analysis manual pdf. Robot structural analysis course. Robot structural analysis system requirements. How to use robot structural analysis.

No textual content provided, but rather a technical manual excerpt. However, assuming a request to paraphrase or reformat the given text, I'll attempt to apply the rewriting methods with a probability-based selection. Rewritten Text (SE: Add Spelling Errors): Looking at Carte 96 04/00 Loads menu / Special loads / Wind Opening of the Snow and Wind 2D/3D dialog box and Snow 2D/3D Press the Auto button; Automatic generation of the strucutre envelope for inactive options: the generation of snow/wind loads (in the Envelope field the without parapets program introduces the folloing node numbers: 1, 2, 3, 4, with base not on ground 10, 9, 13, 12, 11) and definition of basic paramters for the isolated roofs strucutre envelope Define the following paramters: Definition of the basic paramters of snow/wind loads Total depht = (60) Bay spacing = (10) active options: wind snow Press the Paramters button Opening the additional dialog box (Snow/Wind Loads 2D/3D), where one can define detailed paramters Define the paramters of snow/wind Definition of global paramters load: Global paramters tab: Departament: Alpes-Maritimes altidue above the sea level: (200) strucutre height: (15) m reference level: (0.8) m rise of roof: automatic… To access detailed analysis, navigate to the NTM tab and select MY Moments option from the bottom left corner. This will open a graphical viewer with two sections: a graphical presentation of information for selected bars and a table displaying numerical results. In the Detailed Analysis dialog box, choose the quantities to be presented for the selected beam by selecting specific options on the Stresses and Division points tabs. Then, apply these selections and refresh the viewer to add new quantities to be shown for the chosen beam. The Global extremes tab can also be activated to display global extremes obtained for the selected beam. To start a global analysis of all bars in the structure, go to Results menu > Global Analysis > Beginning of global analysis. Another graphical viewer will appear with two sections: a graphical presentation and a table displaying numerical results. In this viewer's context menu, select Table Columns to open the Parameters of Presentation dialog box. On the Stresses tab, uncheck all boxes, then on the Design tab, activate the Ratio option and enter 1.0 as the upper limit value. Next, in the same context menu, choose Constant display of limits to present values with horizontal lines in the graphical viewer. Finally, close this graphical viewer once you're done. The process begins by opening two tabs in the Definitions dialog box, where the first group is defined with parameters including structure number, name, member list, and material. The second group follows a similar pattern but for beams. Next steps involve clicking on the List button in the Code section of the Calculations dialog box to open the Group design line in the Code Group Selection dialog box. A selection window appears with the option to select one to two groups; choosing the first two groups initiates the process. Activating an optimization setting allows the program to utilize specific procedures and limit states, ultimately yielding results that consider ultimate loads and weight sections accordingly. An option is also included to save calculations for future reference. Upon completion of these steps, a message appears indicating that the design requires updating due to changes in profiles. This prompts acceptance of the change with a warning, and subsequently triggers an update status display in the RSAP interface. Finally, closing the Code Group Design dialog box leads to the Calculations button, which initiates the code group design and displays a CM66 – Code Group Design dialog box on screen RSAP displays "Results (FEM): available" once you modify the Standard toolbar sections. To recalibrate, click Calculations and select the optimized options in the Re-design dialog box. This will generate a Short results viewer. You'll need to repeat this process until the desired outcome is achieved. To verify member selection, go to the Member verification field and enter 1-15 in the Calculations dialog box. Then, open the Load case list field and input 1-3. Click Calculations to initiate the verification process, which will provide results for specific members. This step isn't required but can be useful. Once you've completed the member verification, close the dialog box. To customize your printout, access the Printout Composition - Wizard dialog box via File Menu / Printout Composition. Here, you can design the layout of your printout for the current structure. Select the Simplified printout tab to start customizing. To set up the Robot™ Structural Analysis Professional 2010 software, follow these steps: Switch off the options to display quantity survey and combinations data in the printout. Select the following data from the available lists: structure calculations, reactions - global extremes, displacements - envelope forces - values, and stresses - envelope. The Save template button is used to create a new template and go to the Templates tab in the Printout composition - Wizard dialog box. When pressed, it includes the selected simplified printout elements in the right panel. In the Standard tab, highlight the option for Selection of elements for prinout composition and then select Member Group Design. Click the Add button to include this option in the right panel. Next, click the Preview button to view a preview of the defined printout for the designed structure. From the tree in the dialog box, navigate to Materials > Eurocode Design codes > Steel/Aluminum structures and select EN 1993-1:2005 for steel structure design. In the Loads menu, choose the Loads option. Then, go to Code combinations and select EuroCode for automatic code combinations. Next, define structural axes by clicking Geometry > Axis definition. The Structural Axis dialog box appears on-screen. On the X tab, set parameters for vertical axes, including Position, Number of Repetitions, Distance, and Numbering. Repeat this process for horizontal axes on the Z tab. Once defined, generate the structural axes and close the dialog box. The resulting axes are displayed in a figure below. Finally, apply and close the page. To define members, open Geometry > Properties > Sections and select New section definition. Choose the I-section family and define sections such as IPE 240, HEA 240, and HEA 300. Close the dialog boxes and then open the Bars menu by selecting the Bar type field. The process begins by defining the structural lines with green background color in the LMC Beginning field. This involves selecting the HEA 300 section and creating columns marked with numbers 1 and 3 on the A-C range. The same procedure is repeated for the Bar type field, selecting the Beam type and defining a beam positioned between the structural lines with points labeled as B (in the 2-3 range). Next, the Bars dialog box is closed, and the Display dialog box is opened to view the structure in its entirety. On the Structure tab, the Node numbers option is enabled to define the library structure. The Typical Structures dialog box is then accessed by selecting the Library Structure icon from the Structure Model toolbar. The triangular truss of type 1 is selected, and the Merge Structure dialog box appears. Here, parameters such as length and height can be defined for the truss. The Length L field defines the truss length (which can also be set graphically in the graphical viewer), while the Height H field defines the truss height. The Moments option is released with a value of No, and the Sections tab assigns sections to the truss bars. Upper and lower chords are assigned DCED 90x10, while diagonals and posts are assigned CAE 70x7. Insertion Point Field: Defines the starting node of the truss structure, represented graphically as Node #2 with coordinates (0, 0, 6). The entered data can be modified. OK generates the defined truss and closes the Merge Structure dialog box. Auxiliary Node Addition: Edit menu / Divide opens the Division dialog box. LMC field selects the method for defining the insertion point: * In distance: node - through a coordinate on the bar length * Enter value (3.6 m) in the Distance from the top field The auxiliary node is inserted, and the graphical viewer indicates the bar to be divided. Brackets on Bars: Geometry menu / Additional opens the Brackets dialog box. Attributes / Brackets LMC selects the bracket type (Bracket_ 0.1x1). In the graphical viewer, defines brackets at the beam beginning and end (Node #4). Close Closes the Brackets dialog box. Support Definition: Opens the Supports dialog box. Selects fixed support type (Fixed) and assigns it to Node #1 (bottom node of the extreme column). Selects pinned support type (Pinned) and assigns it to Nodes #3 and #5 (bottom nodes of the remaining columns). Close Closes the Supports dialog box. Geometrical Imperfections: Geometry menu / Additional opens the Geometrical Imperfections dialog box. Attributes / Geometrical Imperfections selects Definition of a new type, which opens the Imperfection definition dialog box. 1. In the Label field, enter a new imperfection type with the During_Assembly deflection value set to 5 cm. 2. Disable Automatic and enable User-defined and Absolute options. 3. Add 5 (cm) for the value. 4. Select the default imperfection type (Automatic), then choose a defined imperfection type from the list for LMC in the field with the list of defined attributes. 5. Move to the graphical viewer, select an imperfection type (it will be highlighted) and define it according to EC3. 6. In the bottom status bar, restore a default set of attribute displays by selecting Loads menu / Load Types. 7. Define a load case nature: dead and standard with DL1 as its name. 8. Select live load case nature and name LL1. 9. Close the Load Types dialog box. 10. Select case no. 1 - self-weight load DL1, then define a uniform load on bars that models weight of wall and roof cladding with values: pZ: {-3} (kN/m). 11. Apply this uniform load to bars in LMC the Apply to field. 12. Define a live load case LL1, select the Node tab and choose Nodal Force load. 13. Load parameters include X: {10} (kN) and Z: {-100} (kN), then add this nodal load to the list of loads provided in the bottom part of the dialog box. The Structure View and Crane Load Indicate (LMC) nodes 6 and 18 are highlighted, prompting the user to close the Load Definition dialog box. 3.1.11 Snow/Wind Load Generation: - Opens the Snow and Wind 2D/3D dialog box. - Pressing Auto generates the external structure envelope for a total depth field of 30 meters. - Pressing Parameters opens an additional dialog box for defining detailed parameters, which will be adopted as default. - Generating snow/wind loads with the selected parameters is initiated. 3.1.12 Automatic Code Combinations Generation: - Opens the Load Case Code Combinations dialog box according to EN 1990:2002. - Selecting Full automatic combinations generates code combinations after static structure calculations by clicking OK. 3.2 Structural Analysis and Result Verification: - Runs calculations and opens the results menu. - The bar stress table is displayed, allowing for viewing of maximal stresses in bars at the bottom of the table area. - Calculating global extremes tab displays results on LMC in the top right corner. - The file menu allows saving the example with a selected name using RTD format. 3.3 Elasto-Plastic Analysis: - An additional analysis for accidental hitting by an overhead traveling crane will be performed, considering plastic range analysis. - 3.3.1 Change of Load Case Definitions: - Opens the Load Types dialog box. - Deletes all load cases using the Delete all button. - Defines a new load case with default nature and standard name DL1. - Closes the Load Types dialog box. Loads menu / Load Definition Opens the Load Definition dialog box. Selects the Nodal Force load. Load parameters, Defines values of the nodal load. X: {120} (kN) Z: {0} LMC the Add button provided in the bottom part of the dialog box Move to the graphical viewer with the Defines the nodal load - it models the accidental load structure view and indicate (LMC) resulting from the overhead traveling crane. node no. 18 Close Closes the Load Definition dialog box. page: 44 Autodesk® Robot™ Structural Analysis Professional 2010 3.3.2 Structural Analysis Analysis menu / Calculations Runs calculations Locate the mouse cursor on the Opens the context menu of the structure view. extreme column (bar 1) so that it becomes highlighted, RMC Object Properties Activates the Bar properties option containing information about bar no.1. The Code check tab Performs the simplified design of the steel bar. As it can be seen, it does not satisfy the conditions of code verification. Close Closes the Bar properties dialog box. 3.3.3 Change of Bar Sections for Elasto-Plastic Analysis Opens the Sections dialog box Select the Bar Sections icon from the Structure Model toolbar. LMC on HEA 240 on the section list Selects the current section Select the New section definition Opens the New Section dialog box with HEA 240 section selected icon. LMC the field next to the Elasto- Switches on the elasto-plastic analysis for the section plastic analysis button selected. A new section name is defined: HEA 240EP Add, Close Defines the section HEA 240EP, closes the New Section dialog box. Move to the graphical viewer with the Changes the section of the indicated bars to HEA 240EP structure view and select (LMC) section. external columns (bars no. 1, 2) Accept the warning of changing the result status to ‘not available’. In the Sections dialog box LMC on IPE 240 on the section list Selects the current section Select the New section definition Opens the New Section dialog box with IPE 240 section selected icon. LMC the field next to the Elasto- Switches on the elasto-plastic analysis for the section plastic analysis button selected. A new section name is defined: IPE 240EP Add, Close Defines the section IPE 240EP, closes the New Section dialog box. Move to the graphical viewer with the Changes the section of the indicated bar to IPE 240EP structure view and indicate (LMC) section. 1) To close the Sections dialog box in Autodesk Robot Structural Analysis Professional 2010, you should click on the 'Close' button. This will shut down the dialog box. The Analysis menu allows you to run calculations and verify results. The Results menu enables you to view node displacements or open other related tables. Specifically, the Global Extremes tab calculates maximum displacements of nodes. To start defining horizontal structural axes, go to the Z tab and enter the following coordinates for successive axes: {0}, then insert {3}, followed by {5} and finally {6.5}. Assign numbering starting from 1 and apply it. This creates defined structural axes that close the Structural axis dialog box. Next, open Member Definition, which brings up the Sections dialog box. Select Bar Selections, and under New sections, choose the I-section group. Define a new section by selecting HEA 200, HEA 260, and IPE 200. Once done, close this box to proceed with Bars. In the BAR TYPE field, select Column properties, then in SECTION, choose HEA 260 from page 48 of Autodesk Robot Structural Analysis Professional 2010. In the Beginning field, which changes background color to green, define columns for structure. Column 1 should be between Grids A1-A3 with coordinates starting at (0,0) and ending at (0,5). Column 2 is between E1-E3 with the same height but different x-coordinates. Lastly, define Column 3 between C1-C2 with a shorter height. After defining columns, close Bars to proceed with Library Structure Definition for roof and overhead traveling crane beam. This opens Typical structures where you can select triangular truss of type 1. Define its dimensions in the Dimensions tab by setting length to {12} and height to {1.5}. Lastly, set the number of fields as needed. To create a truss, perform the following steps: 1. Define the number of fields into which the truss will be divided on the Sections tab. 2. Assign sections to the truss bars: DCED 90x10 for upper and lower chords, and CAE 70x7 for diagonals and posts. 3. In the Insert tab, define the insertion point for the truss with coordinates (0,0,5). 4. Create the defined structure in the indicated place within the model. Next, perform the following steps to release the truss: 1. Open the Releases dialog box by selecting Geometry menu / Releases. 2. Select Pinned as the release type and assign it to the truss bar. 3. In the Releases dialog box, select the Current selection field and highlight the truss bar. 4. The arrows on the highlighted truss bar should point up (indicating the direction of the roof ridge). 5. Close the Releases dialog box. To create a library structure: 1. Select the Library Structure icon from the Structural Model toolbar. 2. Open the Typical structures dialog box and select the rectangular truss of type 3. 3. Define the truss parameters: * Length: 12 (can be defined graphically in the graphical viewer) * Height: 1.0 (can be defined graphically in the graphical viewer) * Number of fields: 8 4. Assign sections to the truss bars: DCED 90x10 for upper and lower chords, and CAE 70x7 for diagonals and posts. 5. Define the insertion point for the truss with coordinates (0,0,2). 6. Create the defined structure in the indicated place within the model. Note that I removed some of the redundant language and reorganized the steps to make them more concise and easy to follow. Let me know if you'd like me to make any further changes! Selecting specific bars within the moving-crane truss and central post can be done by highlighting bars 108, 112, and 116. To delete these bars, press the Delete button on the keyboard while selecting "No" to confirm deletion of only the selected bars. This action will apply to the highlighted bars. To define supports for specific structure nodes, select the Supports icon from the Structural Model toolbar. In the Current selection field, choose "LMC" and then select the Nodal tab. Hold down the left mouse button and select points 1, 3, and 5 in the graphical viewer. This will insert these nodes into the Current selection field. Next, hold down the left mouse button again and select all lower nodes of the columns at the level of structural axis 1. In the Supports dialog box, choose a fixed support type by selecting its icon. Pressing "Apply" will apply this selected support to the chosen structure nodes. Finally, close the Supports dialog box to finalize the process.